NASA TM-87021

NASA Technical Memorandum 87021

NASA-TM-87021 19850018953

A Computer Analysis Program for Interfacing Thermal and Structural Codes

Robert L. Thompson
Lewis Research Center
Cleveland, Ohio

and

Robert L. Maffeo
General Electric Company
Cincinnati, Ohio

Prepared for the
International Computers in Engineering Conference and Exhibition
sponsored by the American Society of Mechanical Engineers
Boston, Massachusetts, August 4–8, 1985

NASA

A COMPUTER ANALYSIS PROGRAM FOR INTERFACING THERMAL AND STRUCTURAL CODES

Robert L. Thompson
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio

and

Robert J. Maffeo
General Electric Company
1 Neumann Way
Cincinnati, Ohio

E-2571

ABSTRACT

A software package has been developed to transfer three-dimensional transient thermal information accurately, efficiently, and automatically from a heat transfer analysis code to a structural analysis code. The code is called three-dimensional TRansfer ANalysis Code to Interface Thermal and Structural codes, or 3D TRANCITS.

TRANCITS has the capability to couple finite difference and finite element heat transfer analysis codes to linear and nonlinear finite element structural analysis codes. TRANCITS currently supports the output of SINDA and MARC heat transfer codes directly. It will also format the thermal data output directly so that it is compatible with the input requirements of the NASTRAN and MARC structural analysis codes. Other thermal and structural codes can be interfaced using the transfer module with the neutral heat transfer input file and the neutral temperature output file. The transfer module can handle different elemental mesh densities for the heat transfer analysis and the structural analysis.

INTRODUCTION

In aircraft gas turbine engines, hot section components such as combustor liners and turbine blades and vanes are subjected to severe thermomechanical environments. The cyclic thermal stresses that result are the most important damage mechanism. Consequently, accurate and reliable prediction of thermal loads and the resulting thermal stresses is essential to improving the durability of hot section components. Considerable effort in the past 20 yr has been devoted to the development of heat transfer computer codes for the prediction of steady-state and transient temperatures as well as nonlinear finite element codes for the prediction of cyclic stresses and strains. Continued development of these codes is essential to not only improve durability but also to improve performance and reduce weight. In applying these sophisticated analytical methods to complex three-dimensional engine components, a serious deficiency has developed in the interfacing of output temperatures and temperature gradients from the heat transfer analysis codes with the structural analysis codes. This deficiency has been further exacerbated by the growth in computer capacity and speed and the development of input preprocessors and output post processors. With these advances, the analysis of components using hundreds and even thousands of nodes in the heat transfer and structural models has become economical and routine. To do the interfacing manually or with simple coupling codes for complex models is out of the question.

Although it is recognized that the heat transfer model and the structural model can have the exact same mesh densities, this is not desirable for some applications. For example, in many structural models the areas of high stress gradients may be influenced by thermal loads but are predominantly controlled by geometrical discontinuities such as fillets and holes. A structural mesh to represent the gradients due to these discontinuities would generally be much denser than necessary for the thermal problem. A common mesh might be tolerated for a two-dimensional analysis but many times the additional cost associated with the extra nodes in a three-dimensional analysis cannot be afforded, particularly for transient problems. Another disadvantage of common meshes is the current trend in structural analyses toward adaptive mesh refinement. This technique automatically refines the structural mesh in areas of high stress or strain gradients. Unless these gradients are caused by high thermal gradients there is no need to refine the thermal mesh and redo the thermal analysis just because the structural mesh was refined.

There is also widespread use of finite difference codes to do the thermal analysis, for reasons of convenience, accuracy and/or computer speed. To couple a finite difference thermal code to a finite element structural code necessitates temperature interpolations and thus the need for an interface code. The code could also have the potential to be used for other interpolative applications such as automatic transfer of boundary conditions, for instance, pressures, loads, and displacements.

To meet these needs, a computer code was developed (1-2) to transfer thermal information automatically, accurately, and efficiently from a heat transfer

N85-27264

analysis code to a structural analysis code. The computer code developed is called the three-dimensional TRansfer ANalysis Code to interface Thermal and Structural codes, or 3D TRANCITS. A schematic of TRANCITS is shown in Fig. 1.

MODULAR FLOW OF CODE

The basic program flow is illustrated in Fig. 2. Each of the boxes represents a program module that has been coded to minimize the overall code dependent features. Each module consists of several subroutines. These subroutines, written in FORTRAN 77, communicate with each other through an internal file structure that is hardware independent, easy to follow, and easy to modify. These features not only make the code very flexible and portable but also allow the same basic modular structure to be used in other applications, such as the transfer of other types of loadings. The program flow chart also serves as a basis for discussion of each of the key features of the transfer module. Each of these features is described below in some detail. Also, examples are given to illustrate and to verify these features where appropriate. For a more detailed discussion of the program, see Ref. 1.

INPUT

The user friendliness of the code is provided by the ease of input informational requirements which are grouped into three basic categories. They are user control, geometrical, and thermal data.

The user control data are input to the transfer module with a file that contains all of the variables used to control the interfacing procedure. This file is in namelist format, which allows the user to input the variables pertinent to the problem of interest. Options such as geometry and temporal windowing, element type, coordinate transformations, time step tolerancing, information about the type of input and output desired, and the analysis codes used or to be used are entered through this file. The variables and options are described in detail in the 3D TRANCITS User's Manual (2).

The other two input categories are the geometrical data for both the heat transfer and structural models and the temperature data that are output from a thermal code. The geometrical sizing and identification data for the thermal model and the temperature data are input to the transfer module through what is called a neutral heat transfer input form. This input form supports both finite difference and finite element thermal results. The output data from a heat transfer code and the geometrical thermal data must be compatible with this neutral form. The transfer module does, however, accept the output of the SINDA finite difference and MARC finite element heat transfer codes directly. The SINDA output file, however, does not contain geometrical data, as is the case for most finite difference codes, since volumes, areas, and distances are needed to predict thermal distributions. Therefore, the SINDA output file must be augmented by a SINDA geometry file.

The input associated with the structural model is much simpler than that of the heat transfer model. All that is required is a file containing the name and rectangular Cartesian coordinates of the stress points to be interfaced. If the structural analysis code requires elemental centroid or elemental gauss point temperatures, such as MARC, an additional file containing the stress element connectivity is also required.

FINITE DIFFERENCE CONSIDERATIONS

There are two basic differences associated with using a finite difference heat transfer code rather than a finite element heat transfer code in conjunction with the transfer module. The first, discussed in the previous section, is that in general finite difference codes do not require geometrical data. The second is that the temperatures are produced at the element centroids and at the center of the element faces. Temperatures at these locations are much more difficult to use effectively in a finite element structural analysis.

The approach adopted in the transfer module is to convert centroidal and face-centered element temperatures into accurate temperatures at the vertices of the element. This conversion is accomplished in two steps. The first step is to map a set of face centered and centroidal temperatures of an element to the element vertices. An assumption is made for a three-dimensional linear finite difference volume that the temperatures vary linearly in all three directions between the face centers and the centroid of the elements. Knowing the temperatures at the six face centers and centroid, the temperatures at the eight vertices of the element are determined using three-dimensional tetrahedral isoparametric shape functions. The matrix equation for these mappings is geometry and temperature dependent only and is of the form

$$T = [1,x,y,z][c]^{-1}T \qquad (1)$$

where T is temperature at a node; x,y,z are the coordinates of the node; C is the matrix containing the coordinates of the known temperatures, and the T's are the known temperatures at the face centers and centroid of the element.

The second step is to determine a unique nodal temperature. This is done by weighting each of the nodal temperatures computed at the same location by a scale factor that is inversely proportional to the distance between the centroid of each element adjacent to the vertex and the vertex itself. These steps result in an efficient and accurate prediction of nodal point temperatures.

THREE-DIMENSIONAL SEARCH AND WEIGHTING ROUTINES

To ensure that the interfacing procedure is accurate and efficient, techniques were developed that rapidly determine which heat transfer element contains the stress point of interest. A multistep filtering process was adopted. The initial filter was developed and implemented to eliminate most of the element candidates. It is a coarse filter using a simple algorithm. The possible elements that remain after the initial filtering are then subjected to a more sophisticated check, a fine filter, to determine which element contains the stress point. This search technique is a function of geometry only and therefore must only be performed once, regardless of the number of transient solutions interfaced.

The simple coarse filter requires that the minimum and maximum values of the x,y,z coordinates be stored for each heat transfer element. If the stress point coordinates lie within this element, the element is stored; if not, the search continues. For most models this technique drastically reduces the number of heat transfer elements that could possibly contain the stress point.

The fine filter performs two tasks. It first searches to determine if the stress point lies outside the remaining elements. If it does, the search continues. If the stress point is determined to lie inside the element, the second task performed is to automatically return weighting coefficients that relate the known temperatures of the vertices of the element to the temperature at the stress point location. The weighting coefficients are based on the inversion of the isoparametric shape functions used. The inversion process involves solving for the so-called local isoparametric coordinates based on the global coordinates of the vertices of the element and the stress point. This is shown schematically in Fig. 3.

Basically, the mathematical procedure used in the fine filter is as follows. The equations relating the global nodal point coordinates of a heat transfer element to the global coordinates of a stress point are:

$$X_p = \sum_{i=1}^{n} \alpha_i X_i$$

$$Y_p = \sum_{i=1}^{n} \alpha_i Y_i \qquad (2)$$

$$Z_p = \sum_{i=1}^{n} \alpha_i Z_i$$

where X_i, Y_i, and Z_i are the global coordinates of the vertices of the heat transfer element; X_p, Y_p, and Z_p are the global coordinates of the stress point; n is the total number of vertices; and the α's are the isoparametric shape functions. The first step is to solve for the local coordinates of the stress point (r_p, s_p, t_p) that correspond to the global coordinates of the stress point. The relationships between the shape functions and the local coordinates are well known once the order of the element is known. Substitution of these expressions back into the equations for X_p, Y_p, and Z_p yields a set of nonlinear equations with r_p, s_p, and t_p as the unknowns.

The equations that are solved numerically for r_p, s_p, and t_p are of the form:

$$r_p = f(r_p, s_p, t_p, Xi, Yi, Zi, X_p, Y_p, Z_p)$$

$$s_p = f(r_p, s_p, t_p, Xi, Yi, Zi, X_p, Y_p, Z_p) \qquad (3)$$

$$t_p = f(r_p, s_p, t_p, Xi, Yi, Zi, X_p, Y_p, Z_p)$$

The criteria used for determining if the stress point lies inside the element are:

$$-1 > r_p > 1$$

$$-1 > s_p > 1 \qquad (4)$$

$$-1 > t_p > 1$$

If the stress point lies inside the element, the values of the local coordinates are substituted into the expressions for the shape functions and the thermal weighting coefficients are determined. The temperatures are then determined by the equation:

$$T = \sum_{i=1}^{n} \alpha_i T_i \qquad (5)$$

where the T_i's are the temperatures at the vertices of the element and T is the temperature at the stress point.

A three-dimensional linear isoparametric shape function is used and inverted in the transfer module. However, the technique used can be applied to any order element with a slight modification to the transfer module.

This technique is very efficient and fairly insensitive to a distorted three-dimensional heat transfer element. Several highly distorted eight-node three-dimensional elements were tested to demonstrate the validity of this technique. An example of a highly distorted element is shown in Fig. 4. Although the technique is not sensitive to shape, it is, however, sensitive to the orientation of the element with respect to the global system. In some cases it was found that the technique became unstable and either diverged or converged to an undesirable root of Eq. (3). Logic was added to the transfer module to avoid these stability problems.

A 400-element, eight-noded three-dimensional heat transfer element airfoil model, shown in Fig. 5, was used to validate the search and stability logic. The stress points used were the centroids of the elements and eights points per element that were near the corners of the element. There was a total of 3735 points. After the first transformation, 3728 of the stress points converged. The other seven stress points (orientation dependent) converged after the additional reorientation of the element was completed. These results show that the initial transformation is adequate for most of the stress points and that the additional computation required for the vertex reorientation is only necessary for a very small percentage of stress points.

EXTERIOR POINT SURFACING ROUTINE

To make the transfer module an even more useful tool, stress points that lie outside the heat transfer model can be handled. This was accomplished in 3D TRANCITS by implementing an exterior surfacing routine. This technique makes use of the local coordinates (r,s,t). If the stress point lies outside the element, one of the local coordinates is greater than one. A point that lies on the surface near the stress point would have local coordinate value equal to one. The distance between the stress point outside a heat transfer element and a point on the surface of a heat transfer element can be computed in physical space and stored. The heat transfer element with the smallest distance that can be used is the closest element and the temperature of the exterior point is then determined.

OTHER PROGRAM FEATURES

Other program features include a coordinate transformation that allows the heat transfer model to be aligned with the structural model. There is a capability to "window-in" on a smaller portion of the heat transfer model. There is a provision to select temperature distributions at specific time steps from a large transient thermal analysis. Lastly, error checks are performed and an error file is generated.

OUTPUT

The primary output of the transfer module is a neutral temperature output file. This file consists of the name of the stress node and its corresponding temperature for every transient time point requested. This file can easily be input into simple formatting routines and the stress point temperature configured into any form required by a structural analysis code. The transfer module has the capability to automatically format the temperatures so that they are compatible with the NASTRAN and MARC structural analysis codes. A schematic showing the capabilities for the codes directly coupled to the transfer module is shown in Fig. 1.

VERIFICATION OF THE TRANSFER MODULE

Three different three-dimensional models, from the simple to the complex, were used to verify the isoparametric mapping of the thermal data. The objective was to ensure that the temperatures predicted a the heat transfer code were not degraded in the mapping of the temperatures to a structural analysis model. Comparisons of the results from the heat transfer code to the interpolated temperatures produced by TRANCITS were made using two different methods. The first method used plots to compare the heat transfer results and the TRANCITS results against the model geometry for various regions of the three-dimensional models. These plots were used to verify that the thermal gradients predicted by the heat transfer analysis were accurately mapped into the stress models. The second method for comparing results used the PATRAN postprocessing program. Colored isotherms were created for the stress models using the interpolated temperatures. These isotherms were compared for a qualitative check on the thermal gradients.

The first test case was a simple model of a rectangular prism. The heat transfer mesh was composed of a 2 by 2 by 5 grid; the stress mesh was much finer and had a mesh density of 4 by 4 by 10. These models are shown in Fig. 6. The boundary conditions applied to the heat transfer model give rise to a linear thermal gradient along the length of the prism. The transfer module was used to map these temperatures into the finer stress model. Figure 7 shows a plot of the heat transfer code temperatures and the TRANCITS temperatures versus the distance along the prism. The figure shows that for this simple case the interpolated values agree exactly with those predicted by the heat transfer code. Figures 8 and 9 show the colored isotherms for the heat transfer model and stress model, respectively. Visual comparison of these isotherms again indicate the same thermal gradients in both models.

A more complex model was used for the second test case and is shown in Fig. 10. It is a finite difference model of the tip of a turbine blade composed of ∿450 heat transfer elements. For this case the stress model mesh density was identical to the heat transfer model. The transfer module was used to map the heat transfer temperature at the centroid of the elements and the face centers of the element sides to the vertices of the element. The boundary condition applied to the heat transfer model represents realistic conditions for a turbine blade and gives rise to gradients in all directions. Figure 11 shows a plot of the original heat transfer temperatures and the transferred corner temperatures against a local distance parameter. This plot is for a spanwise section through the airfoil middle section for the convex side of the blade. The local distance parameter is measured from the trailing edge on the convex side. The curves represent the temperatures for a layer of heat transfer elements just above and just below the corner node temperatures. As the plot shows, the corner temperature distribution not only has the same trends as the original heat transfer temperature distribution but also the value of the temperature at each corner node falls in between the temperature above and below the corner as computed by the heat transfer code. These comparisons provide excellent verification of the accuracy of the transferring technique.

Complex models representing the final test case are shown in Fig. 12. These models represent a three-dimensional sector of a combustor liner. The mesh density for the heat transfer model is finer than the stress model in some locations and coarser in others. The boundary conditions used for the heat transfer model once again represent typical engine conditions. Figures 13 and 14 show colored isotherms for both the heat transfer and stress grids, respectively. These isotherms also verify that the gradients are essentially the same for both models.

CONCLUSIONS

The 3D TRANCITS code has the ability to handle different mesh densities for the heat transfer analysis and the structural analysis. It accurately and efficiently transfers the thermal information to produce nodal temperatures, elemental centroid temperatures, or elemental gauss point temperatures for the stress model. Both finite difference and finite element heat transfer analysis codes can be coupled to both linear and nonlinear finite element structural analysis codes. For example, thermal output of both the MARC and SINDA heat transfer codes can be interfaced directly with TRANCITS and it will automatically produce stress nodal point temperatures formatted for NASTRAN and MARC input. In addition to these codes, any thermal code and structural code can be interfaced by utilizing the neutral input and output forms supported by TRANCITS.

Accurate and efficient transfer of temperature data is accomplished in TRANCITS by using all available temperature information to do the interpolation and by using different isoparametric mappings to correspond with different heat transfer elements. Computationally efficient search techniques, consisting of coarse and fine filters, were developed to determine which heat transfer element contains the stress point of interest. Once this is done, accurate weighting coefficients, based on the inversion of isoparametric shape functions that relate the known temperature of the vertices of the element to the desired temperature at the stress point location, are determined and used to predict accurate temperatures at the stress points.

The architecture of the code is such that TRANCITS is both user-friendly and easily modifiable. The code is constructed in modular form so that future modifications, or even different applications (pressure or boundary condition transfer, for instance) can be accomplished without a full rewrite.

For the code to be more useful as an analysis tool, there is a provision to account for stress nodes that lie just outside the transfer model due to slight differences in the dimensions used in the heat transfer and structural analysis models, as a result of using different tolerances on the actual component dimension or a coarse heat transfer mesh compared to a finer structural mesh. There is also a provision that allows the user to select temperature distributions at specific time steps from a large transient thermal analysis. Finally, the technology developed to transfer thermal data forms an excellent foundation for the transfer of other engineering parameters such as pressures, loads, and displacements.

REFERENCES

1. Maffeo, R.J.: Burner Liner Thermal/Structural Load Modeling. NASA CR-174891, 1985.

2. Maffeo, R.J.: 3-D TRANCITS User's Manual. NASA CR-174892, 1985.

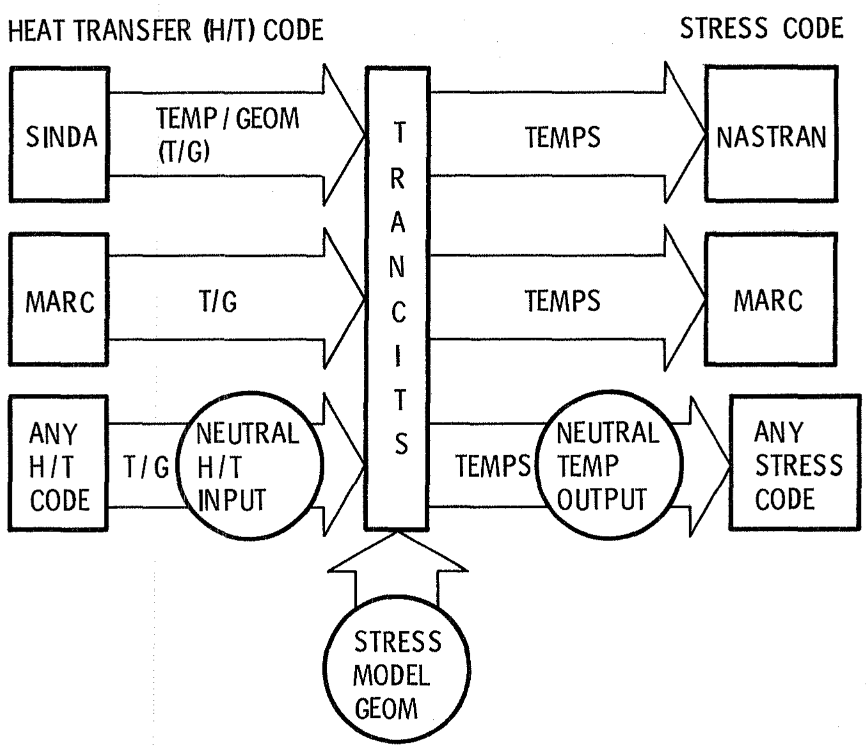


Figure 1. - Overall program schematic for TRANCITS.

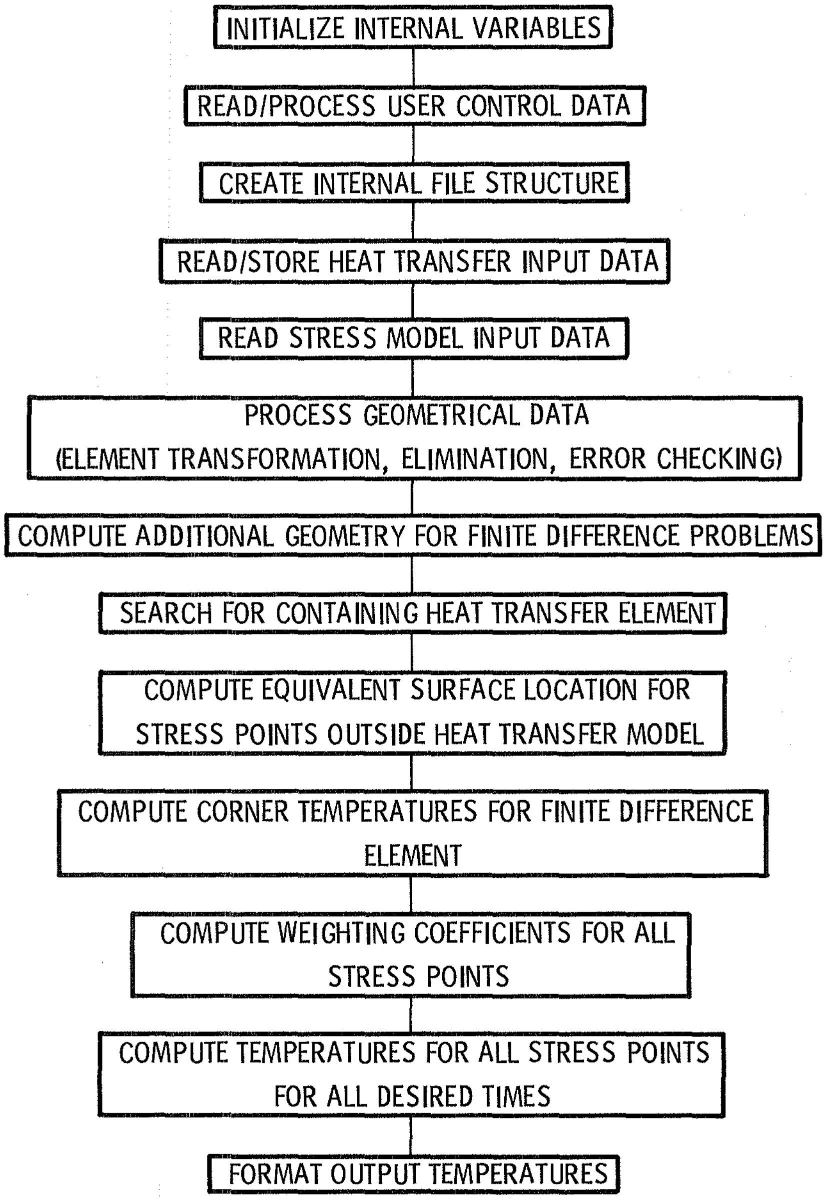


Figure 2. - Modular flow chart for TRANCITS.

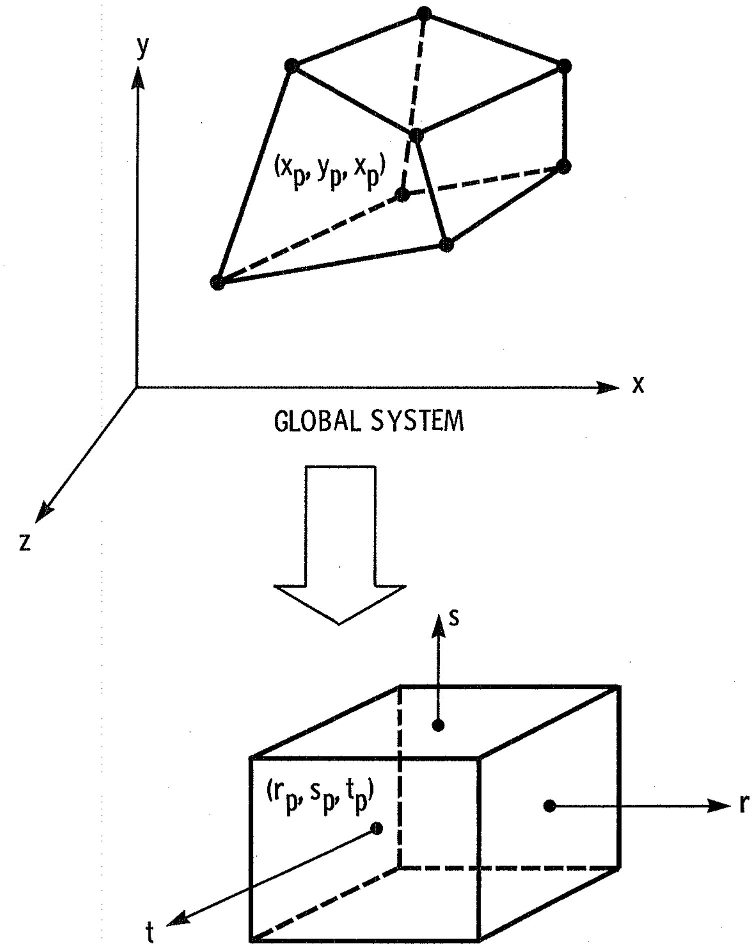


Figure 3. - Global to local isoparametric element mapping.

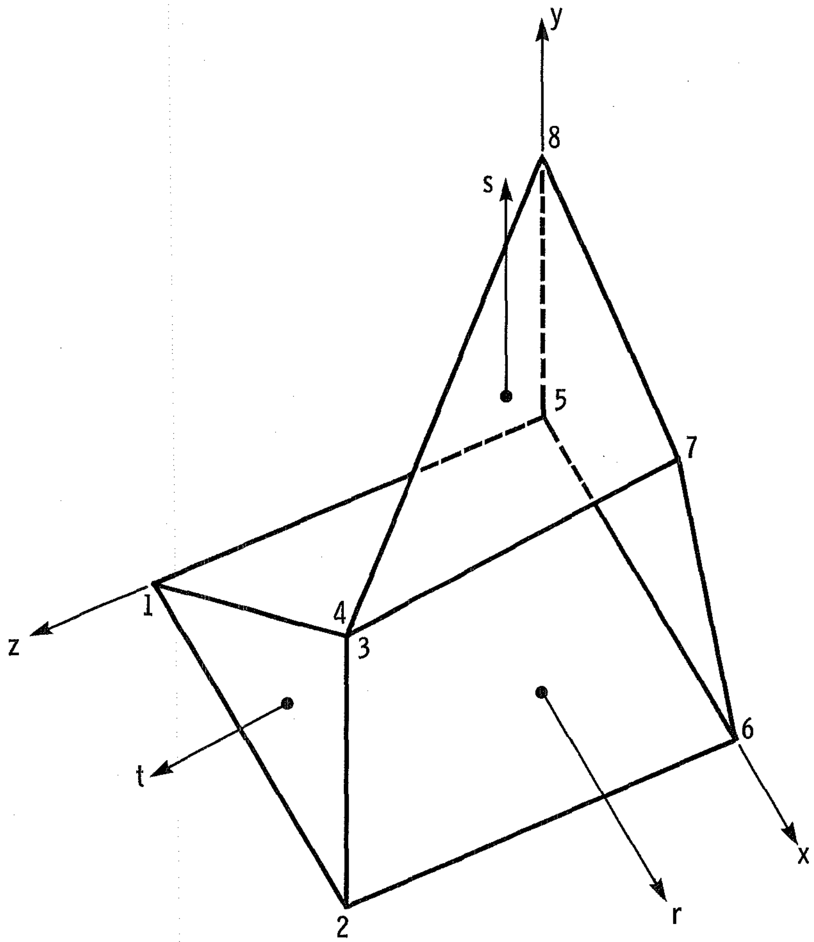


Figure 4. - Highly distorted element with nodes 3 and 4 collapsed.

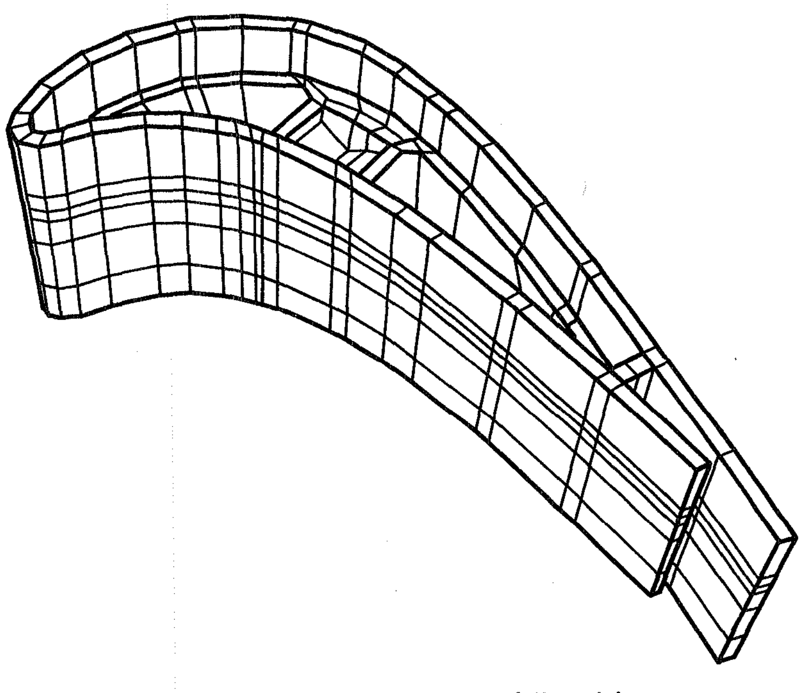
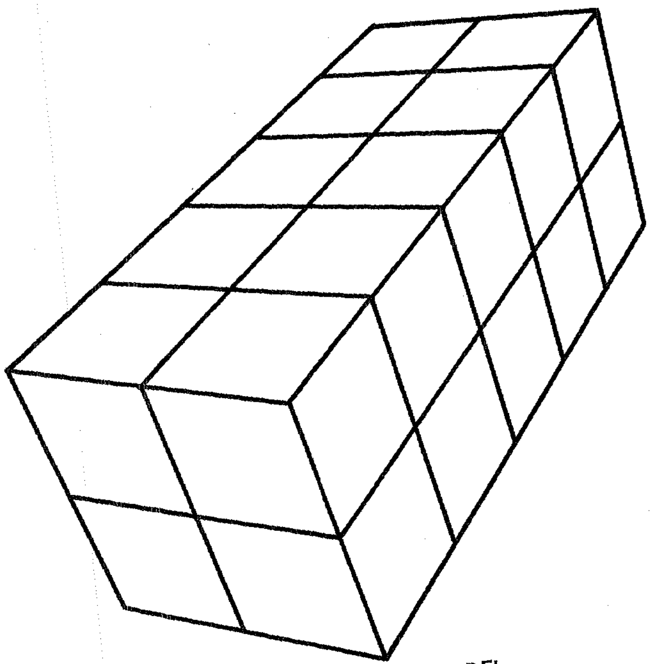
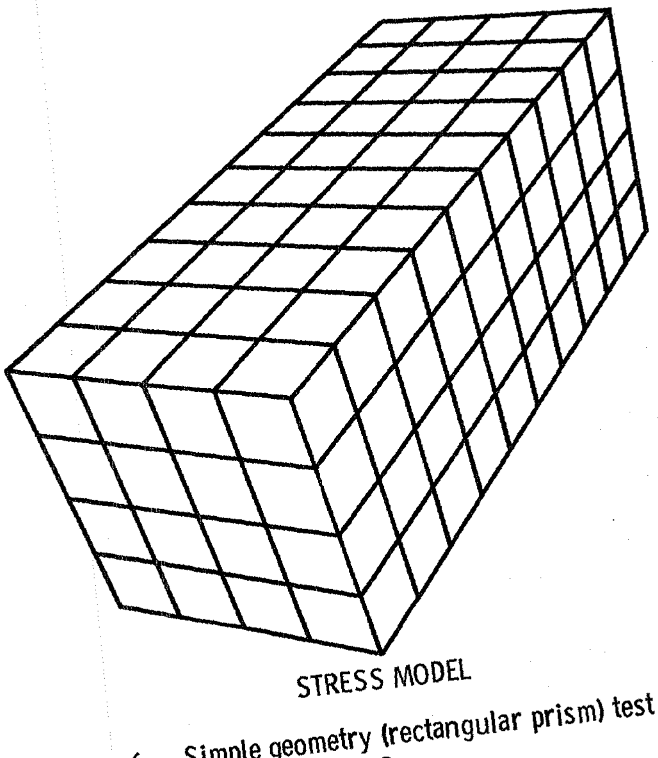


Figure 5. - Heat transfer airfoil model.



HEAT TRANSFER MODEL



STRESS MODEL

Figure 6. - Simple geometry (rectangular prism) test case for verification of TRANCITS.

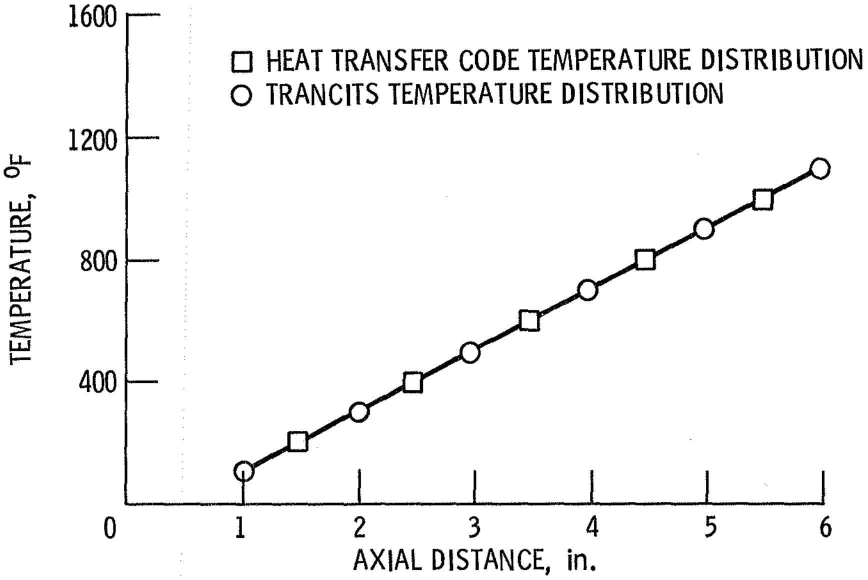


Figure 7. - Temperature vs axial distance for rectangular prism.

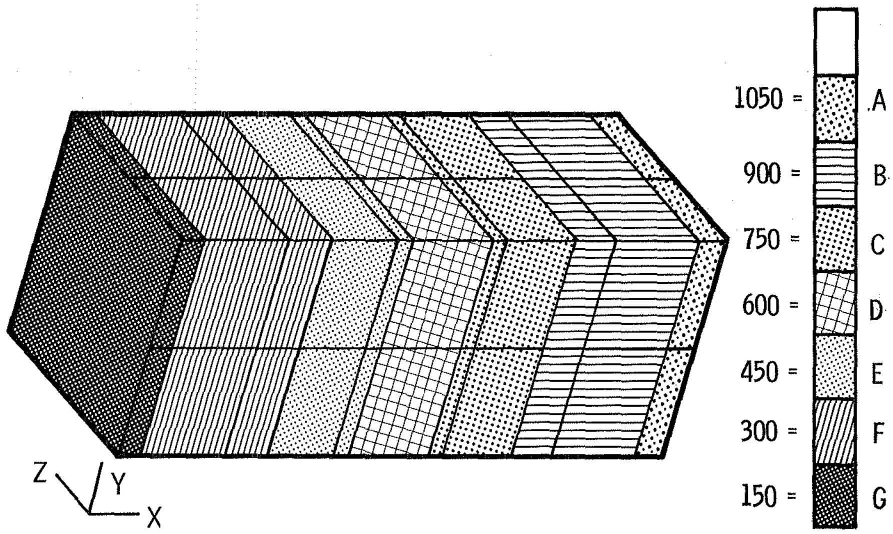


Figure 8. - Isotherms for rectangular prism heat transfer model.

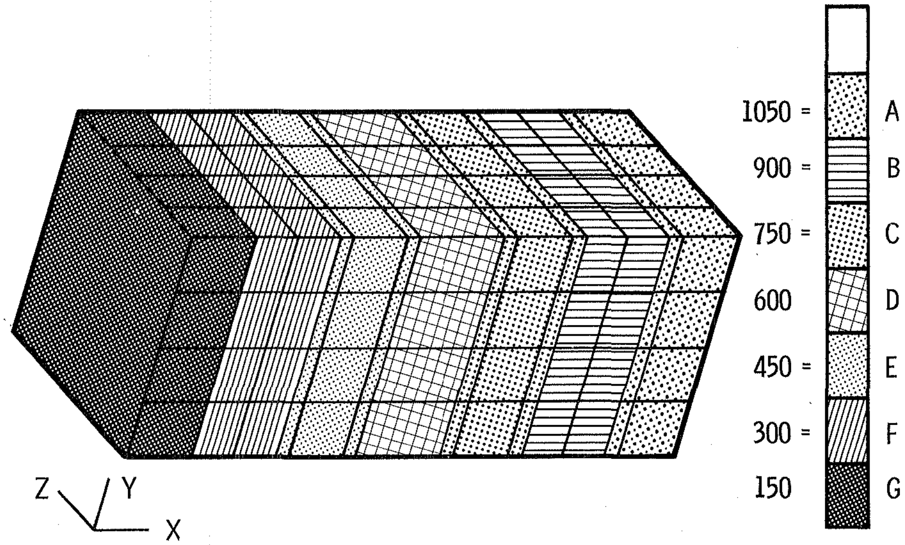


Figure 9. - Isotherms for rectangular prism stress model.

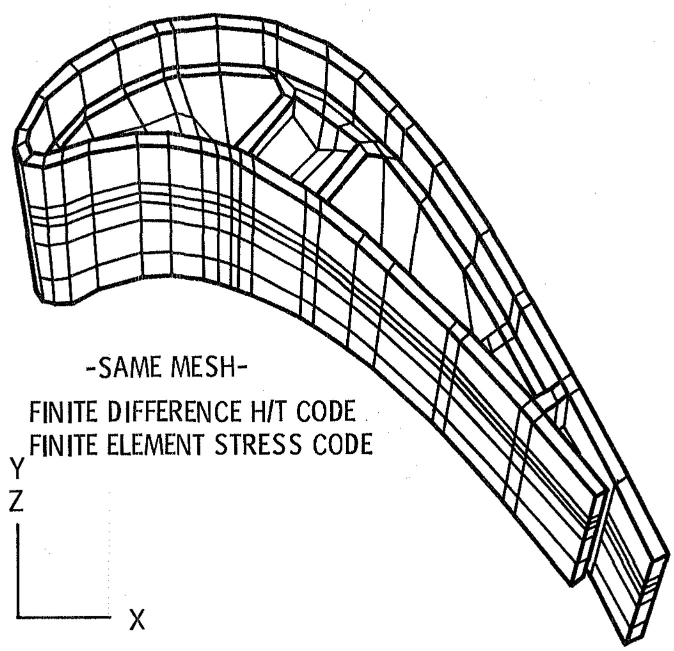


Figure 10. - Complex geometry (turbine blade) test case for verification of TRANCITS.

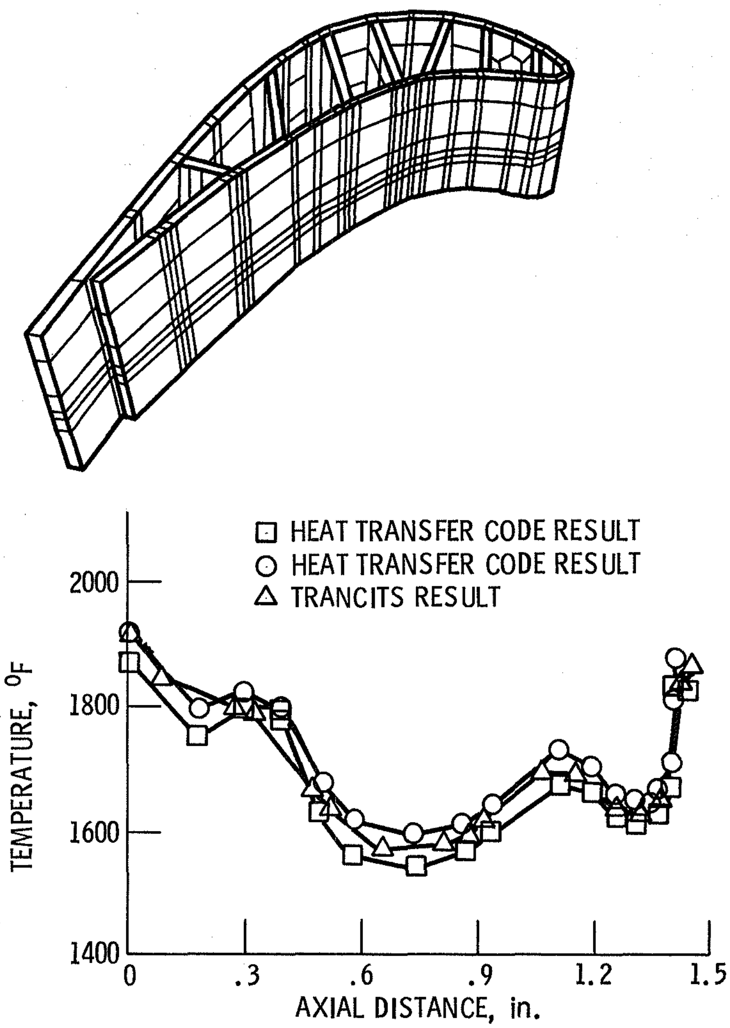


Figure 11. - Temperature vs distance for convex side of turbine blade at the mid-section.

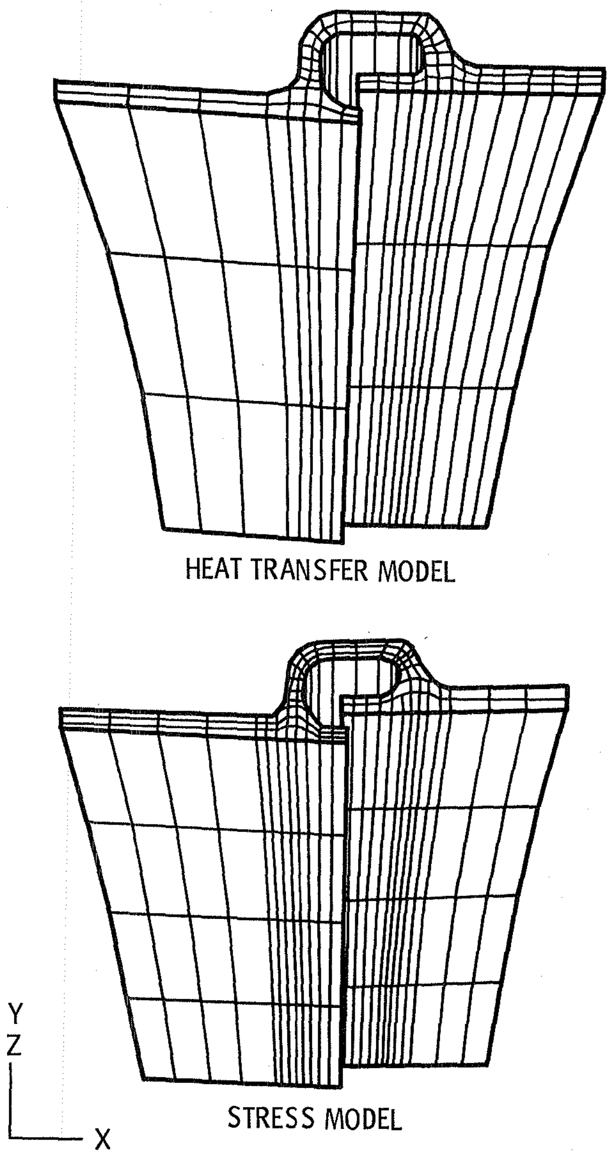


Figure 12. - Complex geometry (combustor liner segment) test case for verification of TRANCITS.

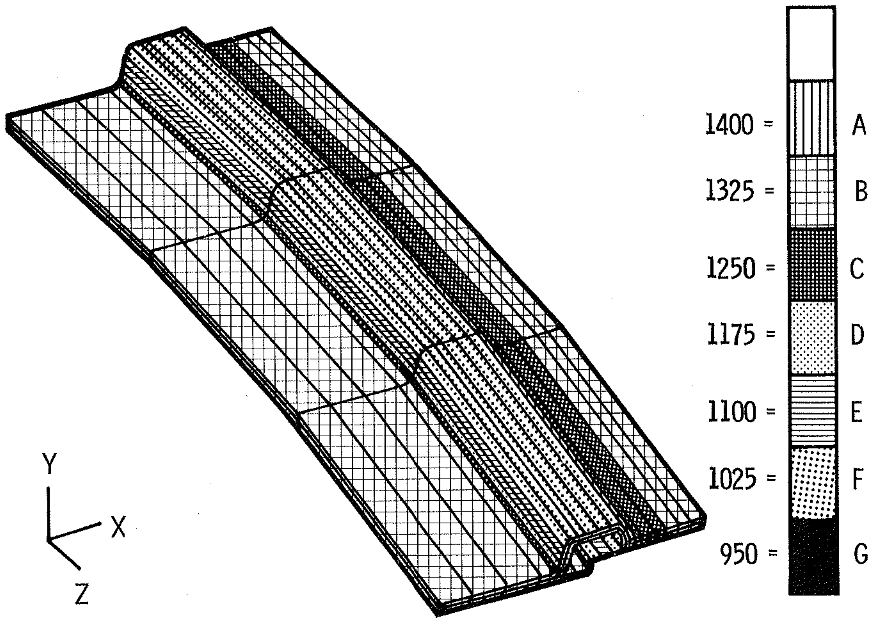


Figure 13. - Isotherms for combustor liner heat transfer model.

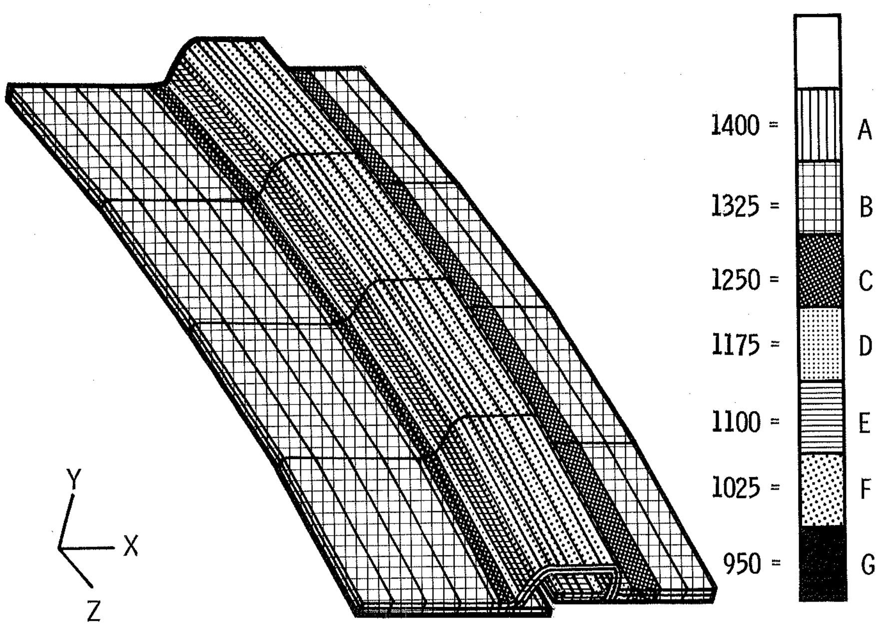


Figure 14. - Isotherms for combustor liner stress model.

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.
NASA TM-87021		

4. Title and Subtitle

A Computer Analysis Program for Interfacing Thermal and Structural Codes

5. Report Date

6. Performing Organization Code
533-04-12

7. Author(s)
Robert L. Thompson and Robert J. Maffeo

8. Performing Organization Report No.
E-2571

10. Work Unit No.

9. Performing Organization Name and Address
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135

11. Contract or Grant No.

13. Type of Report and Period Covered
Technical Memorandum

12. Sponsoring Agency Name and Address
National Aeronautics and Space Administration
Washington, D.C. 20546

14. Sponsoring Agency Code

15. Supplementary Notes
Robert L. Thompson, NASA Lewis Research Center; Robert J. Maffeo, General Electric Company, 1 Neumann Way, Cincinnati, Ohio. Prepared for the International Computers in Engineering Conference and Exhibition, sponsored by the American Society of Mechanical Engineers, Boston, Massachusetts, August 4-8, 1985.

16. Abstract
A software package has been developed to transfer three-dimensional transient thermal information accurately, efficiently, and automatically from a heat transfer analysis code to a structural analysis code. The code is called three-dimensional TRansfer ANalysis Code to Interface Thermal and Structural codes, or 3D TRANCITS. TRANCITS has the capability to couple finite difference and finite element heat transfer analysis codes to linear and nonlinear finite element structural analysis codes. TRANCITS currently supports the output of SINDA and MARC heat transfer codes directly. It will also format the thermal data output directly so that it is compatible with the input requirements of the NASTRAN and MARC structural analysis codes. Other thermal and structural codes can be interfaced using the transfer module with the neutral heat transfer input file and the neutral temperature output file. The transfer module can handle different elemental mesh densities for the heat transfer analysis and the structural analysis.

17. Key Words (Suggested by Author(s))
Heat transfer; Structures; Computer program

18. Distribution Statement
Unclassified - unlimited
STAR Category 39

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of pages	22. Price*
Unclassified	Unclassified		

*For sale by the National Technical Information Service, Springfield, Virginia 22161

End of Document